heaters can be improved by using obstacles in the heating pipes. According to the results obtained in the study, the maximum increase in heat transfer for the obstacle was obtained at a pitch ratio of 2 and an angle ratio of 0.125. Compared to the smooth pipe, this procedure increased the Nusselt number, the friction coefficient, and the thermal performance coefficient by 300%, 69.38%, and 131%, respectively. Compared to the stationary obstacle, the highest thermal performance coefficient for the rotating obstacle was 1.62 times higher at PR = 3 and AR = 0.125 at a rotational speed of 200 RPM.

Key Words: Obstacle rotating, heat transfer, turbulent flow, heat exchanger.

• Received 17 February 2020; received in revised form 7 June 2020; accepted 13 June 2020.

Abstract

The purpose of this study is to investigate the effect of microstructured surface of brass and its morphology change due to the deposition of nanofluids with hybrid base fluid in the pool boiling process on heat transfer characteristics. Pool boiling experiments were carried out with four groups of base fluid: pure deionized water, 70% water - 30% ethylene glycol, 50% water - 50% ethylene glycol, and 30% water-70% ethylene glycol. In addition, nanofluids are made of OH-based multiwall carbon nanotubes with base fluids listed in two volumetric percentages of 0.025% and 0.1%. To the combination of nanofluids used in all experiments was added Sodium Dodecyl Sulfate (SDS) as a homogenizer. In the hybrid fluid pool boiling experiments on the microstructured surface, with increasing ethylene glycol concentration, the critical heat flux and heat transfer coefficient decreased and the highest increase in these two parameters was related to deionized water boiling on the microstructured surface, which was 290.3% and 302.1% respectively, compared to the boiling of deionized water on the polished surface. In pool boiling experiments on the microstructured surface in the presence of nanofluids with hybrid base fluids, the critical heat flux and heat transfer coefficient increased with increasing nanofluid concentration and the highest increase in these two parameters was related to deionized water base nanofluid boiling at a volumetric concentration of 0.1%, which was 74.9% and 91.8%, respectively, compared to the boiling of deionized water on the microstructured surface. On the deposited microstructured surface by nanoparticles, most changes in critical heat flux and heat transfer coefficient were related to deionized water base nanofluid boiling at a volumetric concentration of 0.1%, which was increased by 18.98% and decreased by 48.6%, respectively, compared to the boiling of deionized water on the microstructured surface without deposition. In the all hybrid fluid pool boiling experiments, with an increase in the ethylene glycol concentration, critical heat flux and heat transfer coefficient decreased.

Key Words: Pool boiling, critical heat flux, microstructure, nanofluids with hybrid base fluids, nanoparticle deposition.

EXPERIMENTAL STUDY OF THE EFFECT OF OBSTACLE ROTATING INSIDE HEAT EXCHANGER TUBE ON HEAT TRANSFER AND FLOW CHARACTERISTICS

S.H. Banihashemi
hadi_banihashemi@yahoo.com
Dept. of Mechanical Engineering
Dezful Branch, Islamic Azad University
M.R. Assari(corresponding author)
assari@jsu.ac.ir
Dept. of Mechanical Engineering
Jundi- Shapur University of Technology, Dezful
S.M. Javadi
mohammad.javadi@qiet.ac.ir
Dept. of Mechanical Engineering
Quchan University of Technology
S. Vahidifar
svahidifar@tvu.ac.ir
Dept. of Mechanical Engineering
Shahid Montazeri Technical Faculty
Khorasan Razavi Branch, Technical and Vocational University
DOI:10.24200/J40.2020.54968.1543

Sharif Mechanical Engineering Journal
Volume 36, Issue 2, Page 97-108, Original Article

• Received 19 February 2020; received in revised form 4 July 2020; accepted 18 July 2020.

Abstract

Improving heat transfer in heat exchangers by utilizing different obstacles is of great interest. Moreover, the geometry presented in almost all studies so far has been based on fixed obstacles. Therefore, the current study experimentally evaluates the effects of disk rotation (rotating speeds of 50 to 200 RPM) at different angle ratios and different pitch ratios (1 and 2) on the thermal performance coefficient of the pipe in the air-water heat exchanger with the Reynolds number ranging from 9000 to 26000, and the results are compared and reported. Placing obstacles in the path of the flow is one of the most common techniques for increasing passive heat transfer due to its advantages including easy manufacturing and low maintenance costs. On the other hand, excitation at the boundary layer affects the flow around it as well as the velocity distribution, momentum, and energy transfer. The geometry and location of the obstacle and how it plays an important role in the performance of the system. Various methods for increasing heat transfer in heat exchangers used in the industry as well as its processes to increase efficiency have been studied over the past years due to the importance of optimizing energy consumption and reducing damaging environmental impacts. The results of the current study can also be applied to the oil and gas industry. For instance, the thermal performance of city gas station

have the better performance among all other methods. Using porous media as a new control method, improves power by about 1% in TSR 1.7. Considering the advantages in noise reduction, this new technique may be further investigated. In this paper also parametric study of each control methods have been performed. The change in power coefficient depends on both TSR and actuator intensity. Based on the numerical simulations it is recommended that a hybrid layout consisting active and passive flow control method applied over turbine blades.

Key Words: Vertical axis wind turbine, flow control, riblet, hole, suction, injection, porous media.

STUDYING THE PARAMETERS AFFECTING THE PERFORMANCE OF THE PERISTALTIC PUMP AND DESIGNING A SAMPLE

M.S. Fathi
mfathi.751@gmail.com
A. Hakkaki Fard(corresponding author)
ahakaki@sharif.edu
Dept. of Mechanical Engineering
Sharif University of Technology
DOI:10.24200/J40.2020.55035.1544

Sharif Mechanical Engineering Journal
Volume 36, Issue 2, Page 73-82, Original Article

- Received 12 February 2020; received in revised form 26 June 2020; accepted 18 July 2020.

Abstract

In this paper, first, the general idea of the peristaltic movement is introduced. Peristaltic movement is basically the contraction and expansion of a wall or pipe to move a fluid. Then, the peristaltic pump and its applications and the history of its development are introduced. Peristaltic pumps have many applications in the industry. Then, the movement of the fluid in the pipe of a peristaltic pump is investigated. To this end, the governing equations of a peristaltic pump are introduced and the considered assumptions are presented. Then, the solution to the Navier - Stokes equations (in the steady-state condition) is obtained by simplifying the governing equations, considering the involved boundary conditions. The obtained results are non-dimensionalized. Based on the obtained solution, the relationship of the parabolic radial velocity distribution inside the pump and the mass flow rate of the pump is determined. Then, based on the obtained solution, the tensile pressure of the pipe shell, the pressure applied to the wall by the pump roller, and the fluid pressure are obtained. Next, the relation between them is introduced by considering the equilibrium state between the forces applied to the tube shell. By considering the maximum fluid pressure of a sample pump, a peristaltic pump is designed for pumping water. Moreover, the flow and pressure diagrams and fluid velocity distributions at different tube compression positions of the designed pump are presented and discussed. As a result, the relationship among the pipe compression, the fluid pressure, and the tensile pressure of the pipe is well defined. Furthermore, the relationship between the flow rate of the pump and the speed of its rollers is presented. The linear relationship between the flow rate of the pump and the speed of its rollers is demonstrated and discussed. The volume of pumped fluid per minute is also calculated by determining the velocity of the rollers. The discharge flow from a pump (water fluid) with four rotating rollers is calculated about 0.1 liters per minute. In addition, a flow chart is presented to facilitate the procedures of designing peristaltic pumps.

Key Words: Peristaltic, tube compression, fluid velocity, flow rate, fluid pressure.

EXPERIMENTAL STUDY OF THE EFFECT OF SURFACE TYPE ON THE POOL BOILING HEAT TRANSFER CHARACTERISTICS IN THE PRESENCE OF NANOFLUIDS WITH HYBRID BASE FLUID

A. Dallali
armindallali@gmail.com
M. Khayat(corresponding author)
mkhayat@srbiau.ac.ir
M. Zareh
m.zareh@srbiau.ac.ir
Dept. of Mechanical Engineering
Science and Research Branch, Islamic Azad University
DOI:10.24200/J40.2020.55240.1546

Sharif Mechanical Engineering Journal
Volume 36, Issue 2, Page 83-95, Original Article

M.R. Malayeri(corresponding author)
malayeri@shirazu.ac.ir
Dept. of Chemical and Petroleum Engineering
Shiraz University
DOI:10.24200/J40.2020.54456.1531

Sharif Mechanical Engineering Journal
Volume 36, Issue 2, Page 51-60, Original Article

- Received 22 December 2019; received in revised form 11 April 2020; accepted 2 June 2020.

Abstract

Concerns about emissions and desire for their reduction have resulted in changes in automotive engine combustion systems. New developed EGR systems require high temperature and high-performance compact heat exchangers as well as capability of operating in harsh environments. Foams offer a large surface area per unit volume as well as high material thermal conductivity which would promote fluid mixing, thereby improving the overall performance of the heat exchanger. More interestingly, it is demonstrated that the foams can be cleaned easily without relying on expensive cleaning techniques. These interesting characteristics, along with recent improvements in foam fabrication methods, have resulted in several research efforts on the use of foams in compact heat exchangers, especially new EGR systems. One of the principal challenges is the deposition of particulate matter mainly as a result of thermophoresis in non-isothermal systems. Accordingly, in the present study, to investigate the significance of thermophoresis in open cellular metal foam, Two-Dimensional (2D) numerical simulations of a channel partially filled with aluminium foam as an EGR cooler were performed by ANSYS FLUENT 16.0 to solve local thermal non-equilibrium equation under clean conditions (maximum driving force of particle transport for thermophoresis mechanism). The attempted foam is made of aluminium and is embedded to the channel, meaning that the conduit is only filled with foam partially to compensate high pressure due to blockage. Thermo-hydraulic performance for a foam with density of 20 PPI is examined under different velocities and thermal gradients. The numerical results are compared with those of experiments. Under non-isothermal conditions, the numerical results confirmed that temperature gradient would be marginal for different thicknesses of aluminium foams because thermal equilibrium has been established, especially at foam and foam-free interface. Provided that maximum driving force for thermophoresis occurs at clean conditions, it is expected that this mechanism would have minimal impact at fouling conditions where particulate matter passes through the channel.

Key Words: Heat exchanger, metal foam, numerical simulation, thermophoresis, fouling.

DARRIEUS WIND TURBINE PERFORMANCE ENHANCEMENT USING VARIOUS FLOW CONTROL TECHNIQUES AND COMPARISON

S. Karimian(corresponding author)
karimian@modares.ac.ir
M. Yadegari
mahdi.yadegari@modares.ac.ir
Dept. of Mechanical Engineering
Tarbiat Modares University
Sh. Teymourpour
sh_teymourpour@auto.iust.ac.ir
Dept. of Automotive Engineering
Iran University of Science and Technology
DOI:10.24200/J40.2020.55142.1545

Sharif Mechanical Engineering Journal
Volume 36, Issue 2, Page 61-71, Original Article

- Received 2 February 2020; received in revised form 11 May 2020; accepted 30 May 2020.

Abstract

In this paper performance of vertical-axial wind turbine has been studied based on 5 different flow control techniques. The active methods selected here are suction and blowing and the passive methods are composed of using riblet, cavity and porous media, which the latter one is a new method. The benchmark turbine selected here is Darrieus type and the performance parameter is considered as power coefficient of the turbine. The geometry, location and depth of the actuators are assumed to be approximately equal in different methods aiming the better comparison. Two-dimensional CFD and flow simulation was carried out in Star-ccm+ software. The governing equations are unsteady Navier-Stokes equations and turbulence model is selected k-ŏ3b5. The numerical result of the original wind turbine has been validated against experimental data. Then for each control method, numerical solution has been performed and being evaluated. The results show that the power coefficient or efficiency could be improved as much as the control method would reduce the wake region. The results also show that the blowing and suction control methods with about 6% improvement for TSR 1.7 and 2.64

programming with exact solution, taking into account time factor of $ 6 and $ 8 per hour. The results of the simultaneous allocation of aircraft and passenger ships indicate the use of ships at distances less than 300 km at a time factor of $ 6 per hour and distances less than 200 km at a time factor of $ 8 per hour; also, a comparison of the two optimization methods used for network design shows that the design of the tourism network by integer programming and the hub-and-spoke network design by the minimum spanning tree will be efficient.

Key Words: Minimum spanning tree, air transport network, maritime transport network, combinational network, linear programming.

INVESTIGATION OF NANO-CLAY PARTICLES ADDITION ON MICROSTRUCTURE, FRACTURE SURFACE, AND MECHANICAL PROPERTIES OF PISTON ALUMINUM ALLOY IN AUTOMOTIVE ENGINE

M. Azadi(corresponding author)
m_azadi@semnan.ac.ir
H. Bahmanabadi
h.bahmanabadi1992@semnan.ac.ir
J. Torkian
jahangir.torkian@gmail.com
Gh. Nosrat
ghazaleegh_74@yahoo.com
Dept. of Mechanical Engineering
Semnan University
DOI:10.24200/J40.2020.54945.1542

Sharif Mechanical Engineering Journal
Volume 36, Issue 2, Page 41-49, Original Article

- Received 12 December 2019; received in revised form 2 April 2020; accepted 2 June 2020.

Abstract

This research investigates the effect of the addition of nano-clay particles on the microstructure, fracture surface, and mechanical properties of the piston aluminum alloy in the automotive engine. To this end, aluminum samples with and without nano-particles were cast in stir and gravity conditions, respectively. Then, these initial cylinders were machined to have standard samples for tensile testing. Uniaxial tensile experiments were carried out in the displacement-controlled condition at a displacement rate of 1 mm/min for both reinforced and unreinforced aluminum specimens. Microstructure observations by the optical microscopy showed that the size of silicon particles increased and the size of intermetallic phases decreased due to the addition of nano-clay particles into the aluminum matrix. In addition, according to the uniaxial tensile test, the yield stress, the ultimate tensile stress, and the elongation of samples, made of the base metal, were enhanced. However, the Young modulus of specimens of the base metal decreased due to the addition of nano-clay particles. The fracture surface by the field emission scanning electron microscopy indicated that the reinforcement of the base material by nano-clay particles increased the number of cracks and decreased the total length of cracks. The mapping analysis by the Energy-dispersive X-ray spectroscopy, which was obtained from the fracture surface of the samples, implied that the crack initiation and the propagation occurred at the area, where intermetallic phases were located. Then, the addition of nano-particles into the aluminum matrix had no significant effect on the initiation location and the propagation of micro-cracks. Further investigations into this topic were done on high-cycle and low-cycle fatigue properties of both reinforced and unreinforced aluminum samples to find the effect of the addition of nano-particles to the aluminum matrix, which will be appeared in futures. Finally, it should be noted that the overall objective was to improve the aluminum alloy with nano-particles to have higher mechanical and fatigue properties in order to be used in high-performance engines.

Key Words: Aluminum alloy, automotive engine piston, nano-clay particles, mechanical properties, microstructure, fracture surface.

INVESTIGATION OF THERMOPHORESIS IMPACT ON FOAMED HEAT EXCHANGERS USING SIMULATION

M.H. Pourezaei
pourrezaei1991@gmail.com

• Received 3 December 2019; received in revised form 6 May 2020; accepted 13 May 2020.

Abstract

The present study investigates the speed effect of rotational rotors on the performance of contra-rotating axial compressor and flow behavior. In the present study, a two-stage contra-rotating axial flow compressor with rotor-stator-rotor configuration is considered. The results are presented based on numerical viscous flow simulation in three dimensions by solving Navier-Stokes, continuity, and energy equations using Ansys CFX commercial software. Initially, to validate the results, the absolute and relative flow angle curves for each rotor in the radial direction were extracted using the present simulation and compared with the other investigations. To investigate the effect of different rotor speeds, five different speeds of the second rotor are considered. In the first and second cases, the second rotor speed is reduced by 20 to percent compared to the reference case (in the third case, the speeds of the first and second rotors are equal) and in the fourth and fifth cases, the speed of the second rotor is increased by to percent. The results show that increasing the second rotor speed in the fourth and fifth cases decreases the vortex flows, the blockage in the blade passage and consequently, decreases the flow entropy. The compressor performance curve shows a higher mass flow rate and pressure ratio. In contrast, decreasing the second rotor speed results in undesirable behavior in the flow structure even under design and choke conditions, leading to enhanced tip leakage flow and increased flow entropy which in turn decreases pressure ratio and mass flow rate as well. Results show that by reducing the revolution of the second rotor (in Cases I,II), the tip leakage flow is stronger, resulting in the interface of the main flow and tip clearance flow to upstream which creates vertical flows in design and chock conditions as well as stall condition. This occurrence causes an increase in the original losses and entropy in the compressor.

Key Words: Axial compressor, contra-rotating, numerical simulation, performance curve, rotating stall.

OPTIMUM MIXED (MARITIME-AERIAL) ROUTE-PLANNING USING MINIMUM SPANNING TREE AND INTEGER PROGRAMMING

H. Moaddab
h.moaddab@ut.ac.ir
Aerospace Engineering, Faculty of New Science and Technology
University of Tehran
S.M.B. Malaek
malaek@sharif.edu
Dept. of Aerospace Engineering
Sharif University of Technology
A.R. Kosari(corresponding author)
kosari_a@ut.ac.ir
Aerospace Engineering, Faculty of New Science and Technology
University of Tehran
DOI:10.24200/J40.2020.54727.1538

Sharif Mechanical Engineering Journal
Volume 36, Issue 2, Page 31-39, Original Article

• Received 3 December 2019; received in revised form 12 March 2020; accepted 30 May 2020.

Abstract

Based on World Tourism Organization (WTO) statistics in recent 70 years, the number of tourists has increased from 25 million in 1950 to 1 billion and 235 million in 2016. Extending and developing the transport network is one of the main reasons for this growth. A more developed transport network can improve the tourism rate in country which in return can help the economic growth rate. Iran is considered a very rich country in terms of natural and historical tourism resources; moreover, it has a great potential to become one of the main tourism hubs in the region. Meanwhile, in some cities and places, tourism standards have improved; however, in the southern islands of Iran, despite eye-catching attractions, there has not been enough concerns about them and this issue can be improved by developing the current transport network and designing a new one. In this study, based on the system-system approach, the focus is to design and offer different maritime and air transport solutions between some of Iran's islands in Persian Gulf. In this regard, the research aims to minimize the total cost of a combinational trip consisting of both aerial as well as maritime routes. The devised methodology was applied to finding optimal connecting routes among Persian Gulf islands. The method exploits both Direct Operating Cost (DOC) as well as the value of "time factor". In this study, the final network optimization was performed using two methods of minimum spanning tree and integer

effect of depth and length of cracks on the stress intensity factors will be detected by the results of XFEM. Using the results, the effects of various factors such as corrosion area characteristics, crack geometry including crack depth, crack length, and fluctuations in fluid working conditions including work pressure in fatigue crack growth were investigated. In the model 61%, the first mode of the stress intensity factor of the theoretical method with an error of 0.43% compared to the XFEM showed good agreement with the results of the theoretical methods and the XFEM in the field of fatigue crack growth.

Key Words: Pipeline, fatigue crack growth, XFEM, corrosion.

FAILURE AND COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF PLATEN SUPERHEATER TUBES IN A 320 MW POWER PLANT TO PROVIDE AN OPTIMAL LAYOUT DESIGN

A.A. Shami
aliakbarshami@gmail.com
S.E. Mosavi Torshizi
emoussavi@yahoo.com
A. Jahangiri(corresponding author)
a_jahangiri@ymail.com
Dept. of Mechanical Engineering
Shahid Beheshti University
DOI:10.24200/J40.2020.54621.1535

Sharif Mechanical Engineering Journal
Volume 36, Issue 2, Page 13-19, Original Article

- Received 16 November 2019; received in revised form 1 May 2020; accepted 8 June 2020.

Abstract

Boilers are one of the most important parts of the power plants. Superheater tubes have the main role in boilers and are subject to degradation because of many working conditions. According to the pressure and temperature of the working condition of the superheater tubes in boiler, failure is usually reported from the superheater tubes. Therefore, identifying the causes of this failure and its prevention is quite important. The reports on the failure of platen superheaters in several similar boilers whose inlet and outlet header is the same for all tubes indicate that a particular part of the tubes is more vulnerable than the rest in tubes and the failures are concentrated there. Location of the most failures is at the end of the 135-degree bend of the platen superheater. Therefore, this case was investigated at a special 320 MW power plant where there were seven units. The objective of this investigation is to find the cause of the failure and solve the problem for working in a long time with safety. The preliminary examination showed that the damaged point in this group had a higher temperature than the other groups in the same position. To prove the existence of the higher temperature at these points, three methods were used: metallography, oxide layer thickness measurement, and CFD analysis and all the three methods confirm the results. In this paper, different solutions were introduced for the 320 MW power plant issue. Each of the solution methods is subject to certain advantages and disadvantage. Following the analysis, among the solution methods, one of them was selected as the best solution. Based on the method, some schemes were offered as suggested plans as an alternative to the platen superheater tubes. These schemes were validated in comparison to the power plant superheater. Then, the investigation focused on the geometry of schemes. In addition, all of the three schemes were analyzed by CFD method. Based on the analytical results, one of the schemes was chosen as an alternative plan to the platen superheater for the power plant.

Key Words: Failure analysis, platen superheater, oxide layer thickness, CFD analysis, metallography, thermal analysis,

INVESTIGATION OF THE EFFECT OF SPEED VARIATIONS ON THE CONTRA-ROTATING AXIAL COMPRESSOR STAGES

S. Abbasi(corresponding author)
s_abbasi@arakut.ac.ir
M. Alizadeh
alizadehm59@gmail.com
Dept. of Mechanical Engineering
Arak University of Technology
DOI:10.24200/J40.2020.54868.1541

Sharif Mechanical Engineering Journal
Volume 36, Issue 2, Page 21-30, Original Article

Abstracts of Papers in English

FINITE ELEMENT MODELING OF FATIGUE CRACK GROWTH IN CORRODED PIPELINE

M. Maranlou
maranlou@gmail.com
Dept. of Mechanical Engineering
Science and Research Branch of Islamic Azad University
M. Ganjiani(corresponding author)
ganjiani@ut.ac.ir
Dept. of Mechanical Engineering
University of Tehran
DOI:10.24200/J40.2020.54140.1526

Sharif Mechanical Engineering Journal
Volume 36, Issue 2, Page 3-12, Original Article

- Received 25 August 2019; received in revised form 8 June 2020; accepted 30 June 2020.

Abstract

Ensuring the safety of pipelines in the oil and gas industry is important to prevent financial losses and human injuries. One of the threats of pipelines is corrosion effects and it may occur locally and by repeating changes in loading and operation pressure conditions, it could lead to the phenomenon of fatigue in corroded pipelines. In this research, the corrosion of the inner surface of the API 5L Gr. x52 pipeline using ASME B31.G. and the finite element method was studied and the obtained results along with experimental results were compared. It was found that the effective area method of ASME B31.G. with a maximum error of 3.83% compared to the FEM had good compatibility with the FEM and the laboratory method. Then, by taking into account the phenomenon of fatigue due to working pressure changes, crack growth due to both fatigue and corrosion was studied simultaneously by theoretical methods and extended finite element method. The effect of various factors and conditions on the formation and crack growth due to fatigue by considering the effects of corrosion on the inner surface of the pipeline using XFEM was investigated. By the failure pressure of corroded pipelines, the fatigue crack growth would be investigated and by obtaining stress intensity factors using Paris law and XFEM, results would be investigated in terms of important factors in the fatigue crack growth and the significance and